

A comprehensive comparison of computational methods on propeller modelling of an AUV

Guo Hao Ang
National Centre for Maritime
Engineering and Hydrodynamics
Australian Maritime College
University of Tasmania
Launceston, Australia
guohao.ang@utas.edu.au

Shuangshuang Fan
National Centre for Maritime
Engineering and Hydrodynamics
Australian Maritime College
University of Tasmania
Launceston, Australia
shuangshuang.fan@utas.edu.au

Yuting Jin
National Centre for Maritime
Engineering and Hydrodynamics
Australian Maritime College
University of Tasmania
Launceston, Australia
yuting.jin@utas.edu.au

HuiSheng Lim
National Centre for Maritime
Engineering and Hydrodynamics
Australian Maritime College
University of Tasmania
Launceston, Australia
hui.lim@utas.edu.au

Christopher K. H. Chin
National Centre for Maritime Engineering
and Hydrodynamics
Australian Maritime College
University of Tasmania
Launceston, Australia
c.chin@utas.edu.au

Shuhong Chai
National Centre for Maritime Engineering
and Hydrodynamics
Australian Maritime College
University of Tasmania
Launceston, Australia
shuhong.chai@utas.edu.au

Neil Bose
Bruneau Centre for Research and Innovation
Memorial University of Newfoundland
St. John's, NL, Canada
nbose@mun.ca

Abstract—The propeller of a self-propelled marine vehicle is the key to understanding the hull-propeller interactions. The presence of a propeller not only provides the necessary propulsion force, but also significantly modifies the flow field around the marine vehicle. This modification made to flow field influences the hydrodynamic loading experienced by the marine vehicle and thus has an impact on its maneuvering characteristics. There are various methods to represent the actions of a propeller for a range of marine applications within CFD. Actuator disc or virtual disc is one of the earliest and most commonly used approach. This approach solves the body force generated as an equally distributed force over a cylindrical disc while ignoring the geometry of the propeller. Another method to evaluate the force of a propeller is direct simulations, which replicate a propeller based on its true geometry within CFD. This method utilizes the overset mesh technology within CFD that enables the propeller to rotate independently with respect to the background mesh. The present paper primarily used commercially available CFD software package, Star-CCM+ to investigate the accuracy and feasibility of representing the propeller of an Explorer Class AUV using the earlier mentioned methods. The solved advance ratio, and forward velocity from the two methods was benchmarked against open-water trial data.

Keywords—CFD, AUV, propeller modelling, self-propulsion, virtual disc

I. INTRODUCTION

The propeller of a self-propelled marine vehicle is the key to understanding the hull-propeller interactions. The presence of a propeller not only provides the propulsion force but also significantly modifies the flow field around the marine vehicle. This modification made to the flow field influences the hydrodynamic loading experienced by the marine vehicle and in return influence the vehicle's manoeuvring characteristics [1]. The effects of a propeller on a marine vehicle are commonly studied through either experimental fluid dynamics

(EFD) or computational fluid dynamics (CFD). With the increasing computational power and availability, more researchers have turned to CFD due to its ability to offer a quick solution with high accuracy for its predictions of flow field around a body in fluid and the manoeuvring characteristics of a marine vehicle. Furthermore, CFD methods are more versatile and convenient as compared to EFD.

There are various methods to represent the actions of a propeller for a range of marine applications within CFD, such as actuator disc or direct simulation. Actuator disc or virtual disc method is one of the earliest and most commonly used approach since its introduction in 1865 by Rankine [2]. This approach solves the body force generated as an equally distributed force over a cylindrical disc while ignoring the geometry of the propeller, and due to its computational efficiency, it has often been used for the study of any propeller related marine applications. A recent study by [3] compared the modified Hough and Ordway model against the blade element theory and its respective effects on free turning manoeuvring characteristics for a surface vessel. The mentioned study highlighted a few of the many different models for an actuator disc, and the percentage error between these models against the experimental data are less than 10%. Studies by [4] and [5] have illustrated the robustness of actuator disc in terms of time and cost when applied to the studying of self-propulsion in marine applications.

Another method to evaluate the force of a propeller is direct simulations, which replicate a propeller based on its true geometry within CFD. This method utilizes the overset mesh function within a CFD software package, which enables the propeller to rotate independently with respect to the background mesh. With increasing computational power, this method can solve geometry and flow that are more complicated than before. This approach was used by [6] to perform a series of free-running simulation for the Postdam Model Basin Korean

container ship (KCS) model with independent moving rudder and rotating propeller. The study showed that despite the long computational time, direct simulation can predict the propeller's characteristics and the ship's motion. Furthermore, the direct simulation approach can also provide a more realistic effect of the hull-propeller interactions as illustrated by [7] who investigated the hull-propeller interactions for a submarine at near-surface conditions. In the paper, it was mentioned that the interactions between hull-propeller plays an important role in power performance prediction and the inclusion of a discretized propeller would provide a more realistic geometry of the submarine. In the comparison study by [8] between the direct overset simulation and coupled CFD with potential flow solver for the evaluation of submarine's manoeuvres, it was pointed out that the coupled approach yields less details in the flow field and has higher error.

The above-mentioned articles are some of the many works carried out previously that hopes to provide an accurate representation of a propeller within CFD for the investigation of hull-propeller interaction for a marine vehicle. However, to date, there are limited comparison works carried out for AUVs regarding similar problems within CFD as AUVs behave differently as compared to a surface vessel or a submarine. To this end, this paper aims to provide a comprehensive comparison between two different modelling for an AUV within CFD i.e. virtual disc and direct simulation that may be used for the studying of hull-propeller interactions.

The present study was primarily conducted using the commercially available CFD software package, Star-CCM+. The comparison study investigated the accuracy and feasibility of representing the propeller of an *Explorer* Class AUV using either the actuator disc method or direct simulation method. Thus, to compare the performance of these methods, free running manoeuvring (i.e. straight-line runs, zig-zag manoeuvres and turning circles manoeuvres) were performed and the hydrodynamic coefficients and propulsion characteristics obtained from each run were compared and benchmarked against open-water trial conducted by the International Submarine Engineering (ISE). The open-water curve of the *Explorer* Class AUV's propeller was first predicted through CFD method as its open-water curve was not available in the public domain. The predicted open-water curve data was used to program the virtual disc. In this paper, the comparison of the different propeller representations was based on the advance ratio, and forward velocity, which were obtained from the simulations and validated against the open-water trial. This paper will present the cases when the AUV is travelling in a straight-line. However, hydroplanes were not modelled in all simulations.

II. AUV GEOMETRY

The AMC *Explorer* Class AUV was used throughout this study. The overall configuration of the *Explorer* AUV is rather unique compared to common AUVs. As illustrated in Fig. 2, the AUV has a total of 6 hydroplanes, 2 fore hydroplanes and 4 aft hydroplanes arranged in an X-form and propelled by a twin-bladed Wageningen B-Series propeller. The *Explorer* AUV's and its propeller's principle particulars are listed in Table I and Table II respectively. Although the Wageningen B-series

propeller is a commonly used propeller in the marine industry, the twin-bladed B-Series propeller is used for the *Explorer* that has an area ratio of 0.1, since there are no publicly available open-water curve data for this propeller. Hence, its open-water curve must be predicted.

III. COMPUTATIONAL METHODS

A. RANS Equations

To evaluate the flow around the vehicle, a model based on RANS equations was used. Assuming the fluid is incompressible, the mean flow field governed by the RANS equations can be expressed as [9]:

$$\partial U_j / \partial x_j = 0 \quad (1)$$

$$\frac{\partial U_i}{\partial t} + \frac{\partial}{\partial x_j} (U_i U_j) = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \frac{1}{\rho} \frac{\partial}{\partial x_j} (2\mu S_{ij} - \rho \bar{u}_i' \bar{u}_j') \quad (2)$$

$$S_{ij} = \frac{1}{2} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (3)$$

Where x and t represent space and time, while i and j represent the spatial indexes. ρ and μ represent density and viscosity respectively. $U_{i(j)}$ represent the time-averaged velocity and P is the pressure field. The term S_{ij} refer to the mean strain-rate tensor. The term $\bar{u}_i' \bar{u}_j'$ represent the symmetrical Reynolds stress tensor which consist of six components.

B. 6 DOF and coordinate system

A 6 Degree of Freedom (DOF) motion module was implemented into Star-CCM+ for the AUV hydrodynamics applications. Two set of coordinate systems were used, namely earth system (o', x', y', z') and AUV system (o, x, y, z) as illustrated in Fig. 1.

As illustrated in Fig. 1 for the AUV system, the longitudinal x -axis points aft to fore, the transversal y -axis points from port to starboard, and the vertical z -axis points upwards.

IV. VERIFICATION AND VALIDATION

Verification is the process of assessing the proposed CFD model's numerical uncertainty. Validation is the process of assessing the numerical uncertainty of the proposed CFD model by benchmarking the numerical solution against real data or experimental data. The validation and verification procedure for this study are based on the procedure outlined by [10] with minor simplifications.

The validation and verification studies were conducted for a five-bladed propeller. The study was not carried out for the *Explorer*'s twin bladed propeller, as there was no public available open-water curve. Although, both propellers are different, the methodology to obtain both open-water curves are the same. Therefore, if the methodology to predict the five-bladed propeller is accurate, this would indicate that the same methodology can be used to predict the open-water curve for the twin-bladed propeller.

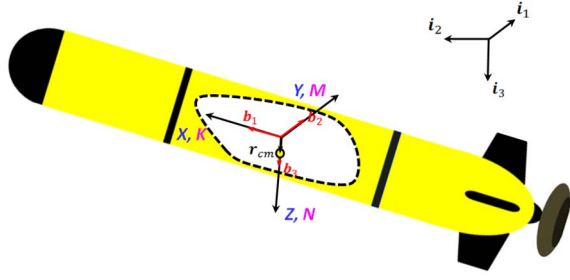


Fig. 1. Reference frame of the AUV in study

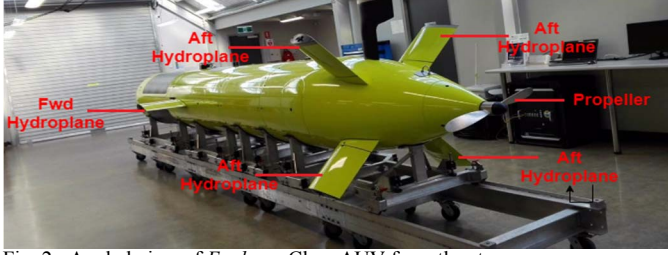


Fig. 2. Angled view of Explorer Class AUV from the stern

TABLE I. Principle particulars of AMC Explorer Class AUV

Parameters	Full-scale dimension
Length (m)	6.6
Diameter (m)	0.74
Slender ratio L/D	8.92
Mass (kg)	1500
Speed (m/s)	1-2.5

TABLE II. Principle particulars of Wageningen B-Series propeller

Parameters	Full-scale dimension
Diameter (m)	0.65
Area ratio	0.1
P/D	0.87
No. of blades	2

A. Open-Water Curve for a five-bladed propeller

Throughout all the runs, the revolution per second (RPS) n was fixed at 13.33RPS while the advance speed was varied accordingly to achieve the desired advance coefficient J . The following formulae were used to obtain the thrust coefficient K_T , torque coefficient K_Q , and efficiency η_0 for each advance coefficient:

$$K_T = T / \rho n^2 D^4 \quad (4)$$

$$K_Q = Q / \rho n^2 D^5 \quad (5)$$

$$\eta_0 = JK_T / 2\pi K_Q \quad (6)$$

$$J = V_A / nD \quad (7)$$

Where T and Q represent thrust and torque respectively, and V_A represents the advance velocity. D is the diameter of the propeller.

The properties of the five-bladed propeller are as outlined in [11].

B. Numerical uncertainty analysis

The numerical uncertainty analysis methodology used in this study was as outlined by [10]. The numerical uncertainty U_{SN} was derived as a combination of grid convergence uncertainty U_G and time-step convergence uncertainty U_T as illustrated in (8):

$$U_{SN} = \sqrt{U_G^2 + U_T^2} \quad (8)$$

Three distinctive set of results are required for each of the analysis. For grid analysis, three mesh sizes with an increment ratio r_i of square root 2 is required. The three mesh sizes can be categorised as fine mesh h_1 , medium mesh h_2 and coarse mesh h_3 . For the time-step analysis, the time-step increment ratio r_i is a fixed value of 2. The three time-step can be categorised as shortest time-step h_1 , medium time-step h_2 and longest time-step h_3 .

The convergence ratio R_i is then defined as the difference in solution S from the three grids or time-steps as follows:

$$R_i = (S_2 - S_1) / (S_3 - S_2) \quad (9)$$

The three possible convergence conditions are:

- 1) $0 < R_i < 1$ Monotonic convergence (MC)
- 2) $R_i < 0$ Oscillatory convergence (OC)
- 3) $R_i > 1$ Monotonic divergence (MD)

For the third condition, grid uncertainty cannot be estimated.

The numerical uncertainty analysis for both grid and time-step convergence study was performed for the case of open-water flow for the five-bladed propeller at an advance speed of 0.8m/s. The results for the grid uncertainty and time-step uncertainty study are summarized in Table III and Table IV respectively.

Through the grid uncertainty study, both torque coefficient and efficiency achieved monotonic convergence while thrust coefficient achieved oscillatory convergence. The uncertainty of the finest mesh size is less than 1% and the associating grid's uncertainty are less than 1% of the finest grid solution. For the time-step uncertainty study, all three variables have achieved monotonic convergence. The uncertainty for the shortest time-step is less than 1% and the associated time-step uncertainty are less than 1%. Therefore, both uncertainty studies suggest that the applied methodology is capable of yielding solutions with reasonably small and acceptable uncertainty.

C. Validating against experimental results

Validation of the simulated results was performed through comparing against the benchmark experimental data from [11]. The validation results are shown in Fig. 3 and Table V.

TABLE III. Grid spacing uncertainty analysis results

Variables	r_G	Solutions			R_G	Convergence	$\delta_G(\%S_i)$	$U_G(\%S_i)$
		S_1	S_2	S_3				
Kt	$\sqrt{2}$	0.3174	0.3170	0.3183	-0.2740	OC	N.A	0.2016
10Kq	$\sqrt{2}$	0.5004	0.5059	0.5202	0.3828	MC	0.3412	0.4197
ETA0	$\sqrt{2}$	0.4039	0.3991	0.3896	0.5116	MC	-0.5063	0.9882

TABLE IV. Time-step uncertainty analysis results

Variables	r_T	Solutions			R_T	Convergence	$\delta_T(\%S_i)$	$U_T(\%S_i)$
		S_1	S_2	S_3				
Kt	2	0.3174	0.3170	0.3183	0.6405	MC	0.0502	0.0942
10Kq	2	0.5004	0.5011	0.5019	0.8212	MC	0.3206	0.8221
ETA0	2	0.4039	0.4037	0.4034	0.9069	MC	-0.1995	0.5575

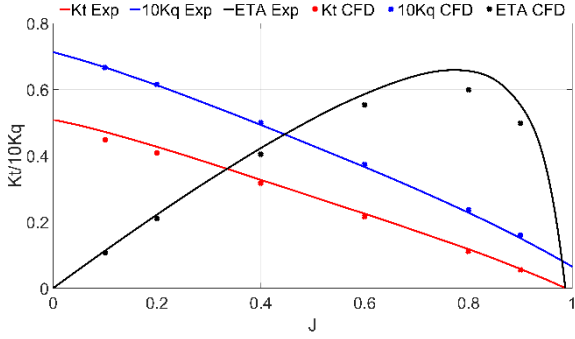


Fig. 3. Comparison between Experimental data (Exp) and CFD Results

TABLE V. Summary of validation results

Variables	$U_{SN}(\%)$	$U_D(\%)$	$U_V(\%)$	$E(\%)$	Status
Kt	0.2016	5.00	5.004	2.902	Validated
10Kq	0.0092	5.00	5.084	1.569	Validated
ETA0	0.0113	10.00	10.064	4.546	Validated

The results illustrated in Fig. 3 show good agreement with the experimental open-water curve data. However, the peak of the efficiency curve was under-predicted. The efficiency of a propeller is evaluated using thrust and torque coefficient as illustrated in (6). Hence, the error in both the thrust and torque coefficient at high advance coefficient were carried forward when tabulating the efficiencies which resulted in under prediction. However, as illustrated in Table V, the percentage error for all three variables are less than 5%, which is acceptable. Hence, this proved that the applied methodology could be used to predict the open-water curve for the Wageningen B-Series twin-bladed propeller.

V. SYSTEMATIC SIMULATIONS

A. Open-water Curve

The open-water curve for the twin bladed Wageningen B-Series used by the *Explorer* AUV was predicted through CFD method. The methodology used for the prediction is similar to the methodology employed previously when determining the open-water curve of the five-bladed propeller. The open-water curve for the twin bladed Wageningen B-Series propeller is as illustrated in Fig. 4. The predicted open-water propeller curve will be used to program all subsequent virtual disc simulations in this study.

B. Virtual Disc

As mentioned earlier, virtual disc solves the body forces generated as an equally distributed force over a cylindrical disc while ignoring the true geometry of the propeller. This approach greatly reduces the computational solving time. Hence, this approach was used to perform five different rotational speeds and its computational results are compared against the open-water trials conducted by ISE in March 2017. The rotational speeds used are listed in Table VI.

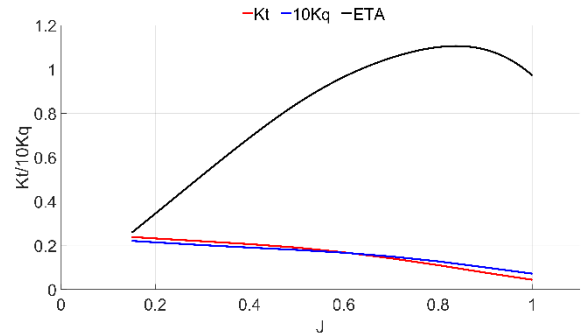


Fig. 4. Twin bladed Wageningen B series open-water curve.

TABLE VI. Rotational speed used for simulations

Runs	Rotational Speed, n (RPS)
1	2.118
2	2.06
3	2.6
4	3.106
5	3.833

These rotational speeds were used during the open-water trials. Furthermore, the selected rotational speed ensured the advance ratio remain constant at approximately 0.74. The main interest of this paper is the period when the AUV travels in a straight line, thus the experimental data were analysed to retrieve the period when the AUV was travelling in a straight line. In addition, the experimental data was processed to remove any environmental forces experienced by the AUV such as underwater current.

C. Direct Simulations

Direct simulations involve modelling the twin bladed propeller of the AUV as it is, that is considering all its features and geometries. Although this method requires longer computational time, it should offer a higher accuracy solution for the investigating problem. Due to the long computational time, only 1 rotational speed (3.106 RPS) was performed and presented in this paper. Likewise, simulation was performed only for when the AUV is travelling in a straight-line with no hydroplanes attached.

VI. RESULTS AND DISCUSSION

The results obtained from the simulations are listed in Table VII. As illustrated from the results, the recorded velocity and advance ratio from simulations without hydroplanes are higher than that from the experimental data, resulting in percentage errors ranging from 23% - 33%. This high percentage error is unacceptable for any CFD simulation.

However, this phenomenon is as expected due to the exclusion of hydroplanes which would significantly change the overall dynamics of the vehicle. The lack of hydroplanes would reduce the overall drag experienced by the AUV hence causing the hydroplanes to travel at a higher speed as compared to experimental data.

However, from the results, it was also observed that there is an anomaly at 3.106 RPS. The percentage error between Star-CCM+ result and experimental result is approximately 5%. Hence this rotational speed was selected to be further investigated using the direct simulation method.

The percentage error obtained is higher than the percentage error obtained through virtual disc simulation of 5%. This higher percentage error is viewed as an accurate representation of the investigated problem due to the lack of hydroplanes present in the CFD model as discussed earlier.

VII. CONCLUSION

Two different methods of representing a propeller within CFD are presented in this paper, 1) virtual disc and 2) discretized propeller. Virtual disc method ignores the geometry of the propeller of interest and solves the body force generated equally over a cylindrical disc. Direct simulation considers all features of a propeller and utilizes the built-in overset mesh function in CFD software to solve the body force generated. Although, the latter method would require a longer computational time, this method would prove to offer higher accuracy solutions when compared to the virtual disc method.

With direct simulation, the lack of public available open-water curve of the Wageningen B-Series propeller was evaluated. The predicted open-water curve was used for all subsequent virtual disc simulations.

A series of rotational speeds was selected from the open-water trial conducted by ISE in March 2017. These rotational speeds were selected after analysing the experimental data to obtain the period when the AUV is travelling in a straight line. This was done to reduce computational time and focus solely on the modelling of propeller for the AUV. The simulated results from the virtual disc method has shown that the percentage error is higher than 20%. However, since all simulations have excluded the effects of hydroplanes, this high percentage error is acceptable and expected. There was an anomaly at 3.106 RPS and hence this rotational speed was selected for the direct simulation. The percentage error obtained is higher than the percentage error obtained from virtual disc simulation of 5%. The higher percentage error obtained will be viewed as an accurate presentation of the investigated problem due to the lack of hydroplanes present in the CFD model.

TABLE VII. Comparison between Experimental data and Star-CCM+ results for simulations using virtual disc

Runs	n (RPS)	J_Exp	Va_Exp (m/s)	J_Star-CCM+	Va_Star-CCM+ (m/s)	Error (%)
1	2.06	0.748	1.016	0.938	1.256	23.62
2	2.118	0.739	1.03	0.938	1.291	25.37
3	2.6	0.746	1.203	0.942	1.592	32.33
4	3.106	0.732	1.806	0.941	1.900	5.193
5	3.833	0.748	1.866	0.943	2.349	25.91

VIII. FUTURE WORK

The current research will proceed to complete the direct simulations. Upon completion, the current research will perform free-running simulations of the AUV through various zig-zag manoeuvres and turning circles manoeuvres cases. In all subsequent simulations, hydroplanes will be included and the hydrodynamic coefficients of the AUV will be evaluated and benchmarked against analytical method performed by ISE.

IX. REFERENCES

- [1] A. Phillips, S. Turnock, and M. Furlong, "The use of computational fluid dynamics to aid cost-effective hydrodynamic design of autonomous underwater vehicles," *Proceedings of the Institution of Mechanical Engineers, Part M: Journal of Engineering for the Maritime Environment*, vol. 224, no. 4, pp. 239-254, 2010.
- [2] W. J. M. Rankine, "On the mechanical principles of the action of propellers," *Transactions of the Institution of Naval Architects*, vol. 6, 1865.
- [3] R. Broglia, G. Dubbioso, D. Durante, and A. Di Mascio, "Simulation of turning circle by CFD: Analysis of different propeller models and their effect on manoeuvring prediction," *Applied Ocean Research*, vol. 39, pp. 1-10, 2013.
- [4] V. I. Krasilnikov, "Self-propulsion RANS computations with a single-screw container ship," in *3rd International Symposium on Marine Propulsors smp*, 2013, vol. 13.
- [5] C. Delen, S. Sezen, and S. Bal, "Computational investigation of self propulsion performance of DARPA Suboff vehicle," *TAMAP J. Eng.*, pp. 1-12, 2017.
- [6] A. Mofidi and P. M. Carrica, "Simulations of zigzag maneuvers for a container ship with direct moving rudder and propeller," *Computers & Fluids*, vol. 96, pp. 191-203, 2014.
- [7] N. Zhang and S. L. Zhang, "Numerical simulation of hull/propeller interaction of submarine in submergence and near surface conditions," *Journal of hydrodynamics, ser. b*, vol. 26, no. 1, pp. 50-56, 2014.
- [8] J. E. Martin, T. Michael, and P. M. Carrica, "Submarine maneuvers using direct overset simulation of appendages and propeller and coupled CFD/potential flow propeller solver," *Journal of Ship Research*, vol. 59, no. 1, pp. 31-48, 2015.
- [9] Y. Jin, J. Duffy, S. Chai, C. Chin, and N. Bose, "URANS study of scale effects on hydrodynamic manoeuvring coefficients of KVLCC2," *Ocean Engineering*, vol. 118, pp. 93-106, 2016/05/15/ 2016.
- [10] F. Stern, R. V. Wilson, H. W. Coleman, and E. G. Paterson, "Verification and validation of CFD simulations," IOWA INST OF HYDRAULIC RESEARCH IOWA CITY 1999.
- [11] O. E. Moctar, V. Shigunov, and T. Zorn, "Duisburg Test Case: Post-panamax container ship for benchmarking," *Ship Technology Research*, vol. 59, no. 3, pp. 50-64, 2012.